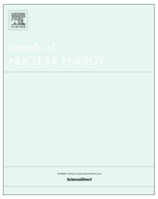
Annals of Nuclear Energy 97 (2016) 190–197

Contents lists available at ScienceDirect

Annals of Nuclear Energy

journal homepage: www.elsevier.com/locate/anucene



# A systematic investigation on flow characteristics of impeller passage in a nuclear centrifugal pump under cavitation state

Qiang Fu [a], Fan Zhang [a,*], Rongsheng Zhu [a], Bo He [b]

[a] *National Research Center of Pumps, Jiangsu University, 212013 Zhenjiang, China*
[b] *POWERCHINA SPEM LIMITED COMPANY, Shanghai 201316, China*

ARTICLE INFO

*Article history:*
Received 24 May 2016
Received in revised form 27 June 2016
Accepted 6 July 2016
Available online 27 July 2016

*Keywords:*
Nuclear centrifugal pump
Flow characteristic
Fluctuation
Cavitation
Numerical simulation

ABSTRACT

Cavitation is a greatly harmful flow phenomenon for nuclear centrifugal pumps, and it should be attached importance when the pump is designed. Under cavitating conditions the flow patterns in the pump are complex and highly turbulent flow can be induced. In this paper the flow characteristics in the impeller passage of a nuclear centrifugal pump model were systematically investigated under steady and transient cavitation conditions. At moderate cavitation levels the results show that the fluctuations of the radial force on the impeller are mainly affected by rotor-stator interaction effects, but are strongly influenced by cavitation under developed cavitation conditions. At inception vapor is mainly generated near the leading edge of the blade, and spreads on the suction side of the impeller at higher cavitation levels. The vapor generation, development and burst under transient cavitation conditions have a strong influence on the flow patterns in the impeller passage. The trends of the simulations are in accordance with the measured results, thus confirming the validity of the numerical model used for predicting the characteristics of the flow through the impeller.

## 1. Introduction

Centrifugal pumps have been widely used in many areas for a long time, but their operational requirements are now becoming stricter and stricter. For example, in centrifugal pumps used in nuclear reactors not only adequate hydraulic performance but also stable and reliable operation under some extreme operating conditions must be attained. The centrifugal charging pumps are parts of the Chemical and Volume Control System (CVCS) and designed to provide normal charging service to the Reactor Coolant System (RCS), which makes them extremely important in nuclear power plants. When the pump operates at a sufficiently low pressure, cavitation is generated, and its development inevitably affects the safety operation of the nuclear pumps. This flow phenomenon takes place in almost all kinds of pumps and has documental effects on the performance of the machine, and therefore should be avoided as much as possible.

In recent years an increasing amount of information is has been made available in the open literature on cavitation in centrifugal pumps. Coutier-Delgosha et al. (2003) numerically and experimentally investigated the performances of a centrifugal pump with two-dimensional curved blades under the cavitating operated condition. Yanxia et al. (2015) used high speed digital movies to study the flow visualization of internal cavitating flow patterns in a centrifugal pump at low flow rates. Xiaojun et al. (2013) analyzed the periodically unsteady pressure field and head-drop phenomenon caused by leading edge cavitation in a single stage centrifugal pump. They pointed out that the vortex flow generation in the rear of the cavitating region is the main reason of the head-drop. Vapor usually is generated and bursts for a short time in the cavitating region of the flow, resulting in the possible onset of flow instabilities caused by cavitation. Many researchers analyzed the flow mechanism of cavitation-induced flow instabilities in order to supply theoretical information for optimal design of pumps operating with better cavitation performance (Tsujimoto, 2001; Yongpeng et al., 2014; Sloteman et al., 2004; Yamamoto and Tsujimoto, 2009; Lee et al., 2009). In addition, a number of other investigations on the cavitating two-phase flow in pumps have been documented in Medvitz et al. (2002), Dular et al. (2005), Zuchao et al. (2008), Poullikkas (2003) and Long et al. (2009).

Transient flow is also a common flow phenomenon occurring in turbopumps under a wide range of conditions, usually during start-up, shut-down and variable load operation, and has been the focus of a number of investigations. Tsukamoto and Ohashi (1982) and Li et al. (2010) studied the instantaneous flow characteristics of centrifugal pumps during the start-up period. Wu et al. (2013, 2010)

---

* Corresponding author.
*E-mail address:* zhangfan4060@gmail.com (F. Zhang).

http://dx.doi.org/10.1016/j.anucene.2016.07.011
0306-4549/© 2016 Elsevier Ltd. All rights reserved.

**Nomenclature**

| | |
|---|---|
| $b_2$ | outlet width of impeller, mm |
| $C_p$ | static pressure coefficient |
| $C_v$ | absolute velocity coefficient |
| $D_1$ | inlet diameter of impeller, mm |
| $D_2$ | outlet diameter of impeller, mm |
| $F_x$ | radial force on X axial direction |
| $F_y$ | radial force on Y axial direction |
| $F_z$ | axial force |
| $F_{cond}$ | condensation coefficient, 0.01 |
| $F_{vap}$ | correction factor of evaporation, 50 |
| $H$ | head, m |
| $K_p$ | pressure coefficient, 5000 |
| $N$ | impeller rotation speed, rpm |
| $P$ | power, kW |
| $p_0$ | pressure when cavitation starts, pa |
| $Q$ | flow rate, $m^3/s$ |
| $Q_d$ | design flow rate, $m^3/s$ |
| $Q_{max}$ | maximum flow rate, $m^3/s$ |
| $R_e$ | steam generation rate |
| $R_c$ | steam condensation rate |
| $R_B$ | bubble radius, m |
| $R$ | radius of impeller, m |
| $T$ | time, s |
| $t_0$ | initial time, s |
| $T$ | temperature, °C |
| $u_2$ | impeller outlet circumferential velocity, m/s |
| $v$ | absolute velocity, m/s |
| $Z$ | number of blades |
| $\Phi$ | wrap angle, deg |
| $\eta$ | efficiency |
| $\alpha_{nuc}$ | nucleation volume fraction |
| $\alpha_{vap}$ | vapor volume fraction |
| $\rho_{vap}$ | vapor density, $kg/m^3$ |
| $\rho_l$ | liquid density, $kg/m^3$ |
| $\Delta t$ | time step |
| $NPSH_r$ | net positive suction head required, m |

investigated the transient flow induced by speed-changes and rapid openings of the discharge valve in centrifugal pumps. Zhang et al. (2014a,b) numerically simulated the transient flow patterns of a nuclear centrifugal pump during changes of its operating conditions at constant rotating speed.

The above investigations only represent a small fraction of the information available in open literature on transient cavitation in centrifugal pumps. However, the adverse effects induced by transient cavitation in pumps are extremely serious, and cannot be ignored in the design process. Therefore, the flow characteristics under both steady and transient cavitation conditions are investigated in our present study, whose results hopefully represent a useful reference for further research work.

## 2. Physical model

The pump model is a centrifugal charging pump, which is an important component of the reactor coolant system of nuclear power plants. Cavitation usually occurs in the first stage of the multi-stage centrifugal charging pumps. Therefore, in the present study, only the stationary domains including the annual suction chamber and the double-channel volute, the rotating domain of the first impeller stage have been selected as the physical model in order to investigate its cavitation characteristics. The main geometric and hydraulic specifications of the pump are reported in Table 1. It is required $NPSH_r \leqslant 7.8$ m according to the CVC system in a nuclear power plant at the maximum flow rate $Q_{max} = 160\ m^3/h$. The flow domains of the centrifugal charging pump with its first stage impeller have been modeled by Pro/E, as shown in Fig. 1.

## 3. Numerical approach

The RANS equations have been performed by using the ANSYS CFX 14.5 software, which uses a multi-block technique to couple the separate numerical domains. A high mesh quality is required in the numerical simulations for improving the precision of the results and reducing the computational time. All parts of the pump model have been meshed with structured hexahedral grids, and the meshes of boundary layers have been thickened. Generally speaking, the calculated error caused by the mesh are reduced by increasing the number of gird elements, but excessive computer memory and running time are required in the numerical simulation if the grid number is too large. Hence the selection of a suitable number of grid elements is very important for the simulation. The predicted head performance of the pump with 4 different numbers of grid elements are compared in Fig. 2 and the detail of various grid meshes are reported in Table 2. In general, the head of the model pump increases with the number of grid elements increases. However, the head obtained for grid number 4 only increases by less than 1 m with the respect to the results of grid number 3, while the head discrepancies for the other grids are significantly larger. Therefore grid number 3 was selected in the present simulations. Fig. 3 shows an overview of partial mesh of the flow passage, especially the main parts of the geometric structure.

The Zwart-Gerber-Belamri model, as expressed by (1) and (2), proved to yield better precision for cavitation simulation (Zwart et al., 2004), has and therefore been selected in the present work.

$$R_e = F_{vap}\frac{3\alpha_{nuc}(1-\alpha_{vap})\rho_{vap}}{R_B}\sqrt{\frac{2}{3}\frac{p_{vap}-p}{\rho_l}}, \quad p < p_{vap} \tag{1}$$

$$R_c = F_{cond}\frac{3\alpha_{vap}\rho_{vap}}{R_B}\sqrt{\frac{2}{3}\frac{p-p_{vap}}{\rho_l}}, \quad p > p_{vap} \tag{2}$$

The SST $k$-$\omega$ turbulence model is a hybrid model combining the advantages of the standard $k$-$\omega$ and $k$-$\varepsilon$ turbulence models. The viscous flow near the wall and the turbulence fully developed turbulent region can be accurately modeled using the standard $k$-$\omega$ and standard $k$-$\varepsilon$ turbulence models, respectively. As a result, the SST $k$-$\omega$ turbulence model was applied in this investigation to solve the RANS functions.

The average static pressure has been specified at the inlet of the suction chamber, with uniform normal flow running into the inlet section. The mass flow rate has been assigned at the outlet of the double-channel volute. The impeller domain rotated with speed

**Table 1**
Specifications of the pump model.

| Geometric specifications | | | Hydraulic specifications | | |
|---|---|---|---|---|---|
| Inlet diameter (mm) | $D_1$ | 140 | Nominal speed (r/min) | $n$ | 4500 |
| Outlet diameter (mm) | $D_2$ | 236 | Design flow rate ($m^3/h$) | $Q_d$ | 110 |
| Outlet width (mm) | $b_2$ | 12 | Maximum efficiency | $\eta$ | $\geqslant$60% |
| Wrap angle (deg) | $\varphi$ | 135 | Maximum flow rate ($m^3/h$) | $Q_{max}$ | 160 |
| Blade number | $Z$ | 4 | Net positive suction head required(m) | $NPSH_r$ | $\leqslant$7.8 |